[논문] 태양에너지
Solar Energy
Vol. 19, No. 1, 1999

Cavity내의 단일 열원에 대한 최적 열적설계

예용택*, 추홍록, 김현우*****

* 경일대학교 기계공학부
** 대구보건대학 산업안전과
*** 안동공업대학 기계공학과

Optimal Thermal Design of a Single Heat Source in a Cavity

Y. T. YAE*, H. L. CHOO**, H. W. KIM***

* *Department of Mechanical Engineering, Kyungil University*
** *Department of Industrial Safety, Taegu Health College*
*** *Department of Mechanical Engineering, Andong Technical College*

ABSTRACT

The optimal thermal design of a single heat source on one wall of a vertical open top cavity was studied experimentally. The temperature and flow fields in the cavity were visualized. The objectives of this study is to obtain the best location of the single heat source and to examine the effects of heat source protrusion, substrate thermal conductivity and cavity aspect ratio on the natural convection cooling due to a single heat source.

As the results, the cooling effect for the copper substrate is superior to that of the epoxy-resin substrate and is improved with increasing cavity width. For the epoxy-resin substrate of lower conductivity, the protrusion of the heaters plays a role in decreasing the cooling effect. The best location was the mid-height of the substrate.

Nomenclatures

A	: aspect ratio, H/W
H	: cavity height
Q_e	: power input per each heat source
T	: temperature
W	: cavity width
y	: local height measured from the bottom of the cavity to the mid-height of heat source

Subscripts

max	: maximum value
w	: vertical wall
∞	: ambient air

1. Introduction

Generally, many practical problems related to the thermal design are occurring in devices such as solar cells and electronic components involves energy.

The thermal design requires that the power system module box be closely coupled to the solar-panel is approximately ten times that normally used. To ensure that all the component of the spacecraft operate within the appropriate temperature bounds, given all the possible thermal environments that the spacecraft may encounter. The thermal environment of a spacecraft hanges drastically in a short time, and it complicates the thermal control of the electronics$^{1)}$.

The present trend in spacecraft design is toward high power electronic amplifiers with very localized heat dissipation and strict component temperature limits. Efficient thermal spreading is needed to distribute the heat over the base area of high power electronic boxes. As the reliability of the electronics is strongly dependent on its temperature, the temperature extremes of the electronic components must be minimized in space applications where a very high reliability is essential$^{2)}$.

Radiation and natural convection are the simplest to use of all the heat transfer method.

In most electronic equipment, heat transfer occurs by radiation and natural convection simultaneously. Therefore, the effective cooling of electronic components has become increasingly important from the view point of equipment stability, reliability, and good operational life. The most desirable and simplest cooling method of components is air circulation by naturally generated buoyant forces. The natural convection cooling plays an important role in several areas of practical concern, such as the cooling of electronic equipment and positioning of heated elements in furnaces.

Several works for the heat transfer from a single heat source on surface were carried out by Oosthuizen$^{3)}$, Jaluria$^{4)}$, Churchill$^{5)}$ and Richards$^{6)}$. The most frequently encountered geometry in their studies is a vertical channel or enclosures. However, a large number of components normally is packaged in the vertical open top cavity. Davis and Behnia$^{7)}$ carried out a numerical study of transient natural convection in a two-dimensional upright cavity open at the top with a flush-mounted single heater. Riu et al$^{8)}$ did an experimental study of transient natural convection in a 2-D vertical open top cavity with multiple discrete heaters on one wall. From the preceding review of pertinent literature, it seems that experimental studies for the natural convection cooling of discrete single heat source in a vertical open top cavity have to be explored.

In the present experimental study, the problem of 2-D steady natural convection in a vertical open top cavity with discrete single heat source is considered. To study flow structure and heat transfer characteristics, the flow and temperature fields in a cavity were visualized using Mach-Zehnder interferometry and smoke-method. The main objectives is to obtain the best location of the single heat source and to examine the effects of the heat source protrusion, substrate thermal conductivity and aspect ratio on the natural convection cooling.

2. Experiment

2.1. Experimental Apparatus

The schematic diagram of the experimental apparatus and the details of the vertical open top cavity are given in Fig. 1.

The present experimental facilities consist of four major parts, i.e., test section, power supply, digital data acquisition and visualization system. The basic facilities used in the present experiment are almost the same as in the experiment made by Riu et al.[8].

The test section is a vertical open top cavity. The power to the heat source was provided by passing an electric current from a D.C. power supplies. The local surface temperatures were measured using a data acquisition system, controlled by a workstation. The temperature field of natural convection within the open top cavity has

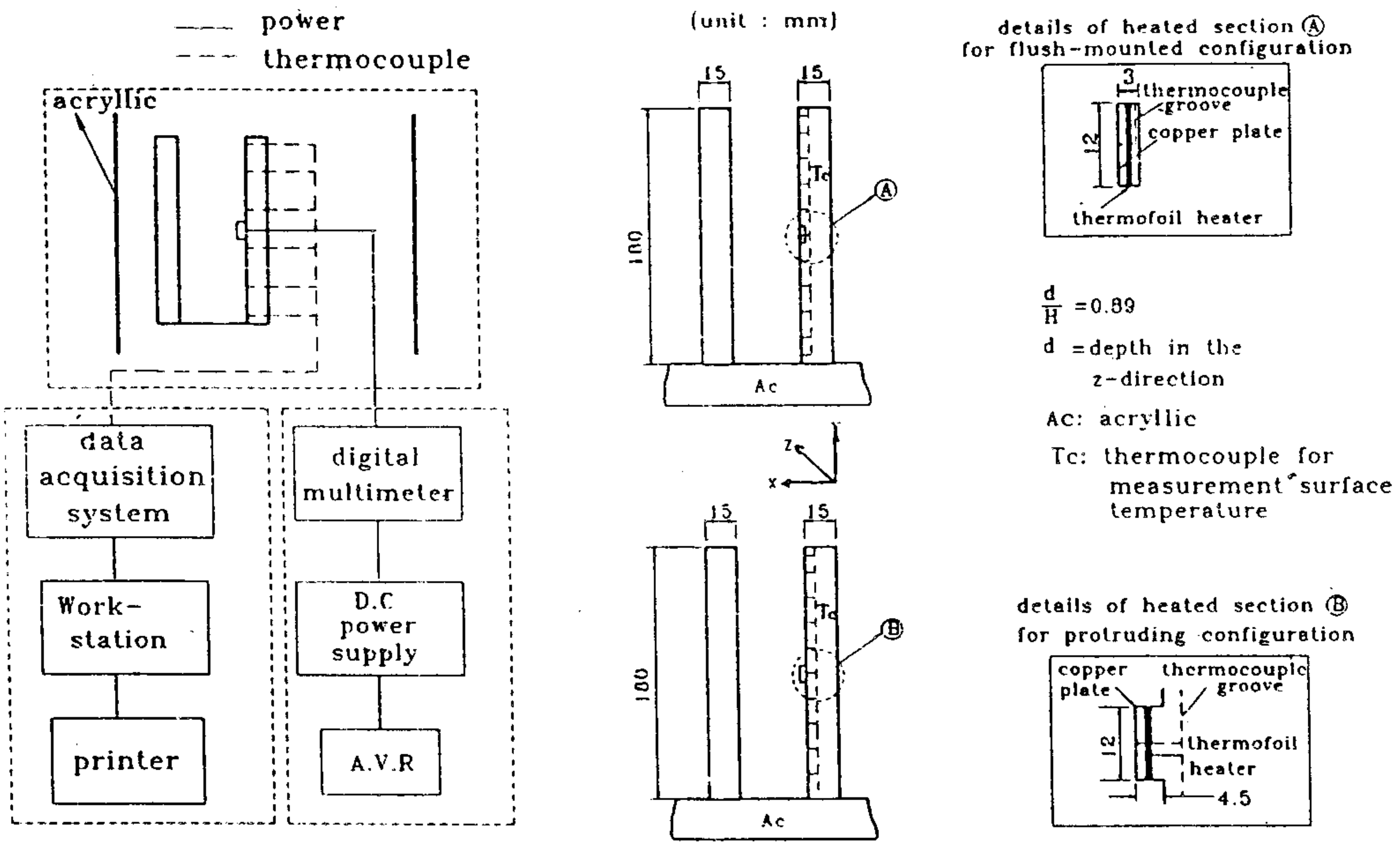


Fig. 1. Schematic diagram of the experimental apparatus and details of the vertical open top cavity

been visualized by means of Mach-Zehnder interferometry with 200 mm optics. The light source used was a 30mW He-Ne laser. To examine the typical flow structure, flow visualization experiments were conducted using smoke-generated from a cigarette as the flow tracer.

The vertical substrates of the cavity, 180× 163.5 mm, were constructed of 15 mm thick copper(K=398 W/mK) and epoxy-resin (K=0.6 W/mK) sheet used as substrate material respectively. The cavity was 163.5 mm in the spanwise direction, ensuring near two-dimensionality of the flow and heat transfer. The discrete single heat source was adhered to the y/H=0.1, 0.3, 0.5, 0.7 or 0.9 for each experimental condition. The vertical walls of the front, rear ends, and horizontal bottom wall of the cavity were made of 15 mm thick acryl to enable optical visualization of flow fields and to minimize the conduction loss.

The heat source modeled electronic component is 3 mm thickness, 12 mm width and 163.5 mm depth (Fig. 1, details Ⓐ). To achieve heat flux uniformity, a rectangular base, 2 mm deep, 10 mm wide and 155 mm long, was cut in copper plate to place 10× 155mm thermofoil heater with a thickness of 0.5 mm, resistance of 22.1Ω. There are a total of 11 calibrated thermocouples(T type, 0.25 mm dia.) installed to measure local temperatures of the test plates. The thermocouples were epoxied to midheight of the heat source and of 10 unheated section on the vertical surfaces. For the vertical open top cavity with protruding heat source configuration, the geometry and the details for the cavity,

heater assembly and thermocouple location correspond with that for flush-mounted heat source configuration. However, the heat source was arranged in configuration protruding from the wall about 4.5mm (Fig. 1. detailsⒷ).

2.2 Experimental Procedure

The preliminary test to assure the heat flux uniformity showed in a ±0.2℃ maximum temperature difference along the heat source surface. The test section was positioned vertically by supporter and adjusted on required cavity width. In addition, output voltage of D.C power supply with a multimeter was regulated to a specified power input. The experiments were carried out by monitoring the temperatures on the heat source until steady state was reached. Once data collection was completed, the visualization experiments for temperature and flow fields the inside of the cavity were conducted by using Mach-Zehnder interferometry and smoke-method.

Data Reduction

The local temperature rises, $\triangle T_w$ and maximum temperature rises, $\triangle T_{w,max}$ are defined as

$$\triangle T_w = T_w - T_\infty \tag{1}$$

$$\triangle T_{w,max} = T_{w,max} - T_\infty \tag{2}$$

where T_w is the local temperature, $T_{w,max}$ is the maximum temperature on the substrate and T_∞ is the ambient air temperature. The analysis for isothermal contour of fringes is given by the Dale-Gladstone equation[9) .

In this experiments, the accuracy of the temperature measurements was ±0.1℃. The voltage input and resistance to each heater circuit were measured with an accuracy of 1% and 0.06% respectively. The uncertainty associated with the length was ±0.25 mm.

The error between the measured temperature by thermocouples and calculated by interferometry was within about 5%. This error might be due to the removal of front and rear walls for visualization work.

3. Results and discussion

The experiments with air as convective medium were conducted for power input of 2.0W, 6.0W, 8.0W, and 19.0W. The two-dimensional cavity used in the study was 180 mm tall with variable widths of 19 mm (A=9.5), 24mm (A=7.5), and 39mm (A=4.6). Two different conductivity of the vertical substrate were tested.

In general, one of the most important objectives of thermal design in electronic packages is to maintain the component temperature below the specified maximum operating temperature. Hence, in the presentation of the results, the results are presented in terms of the local temperature rises and the maximum temperature rises on the vertical substrate.

3.1 Effect of the Substrate Thermal Conductivity

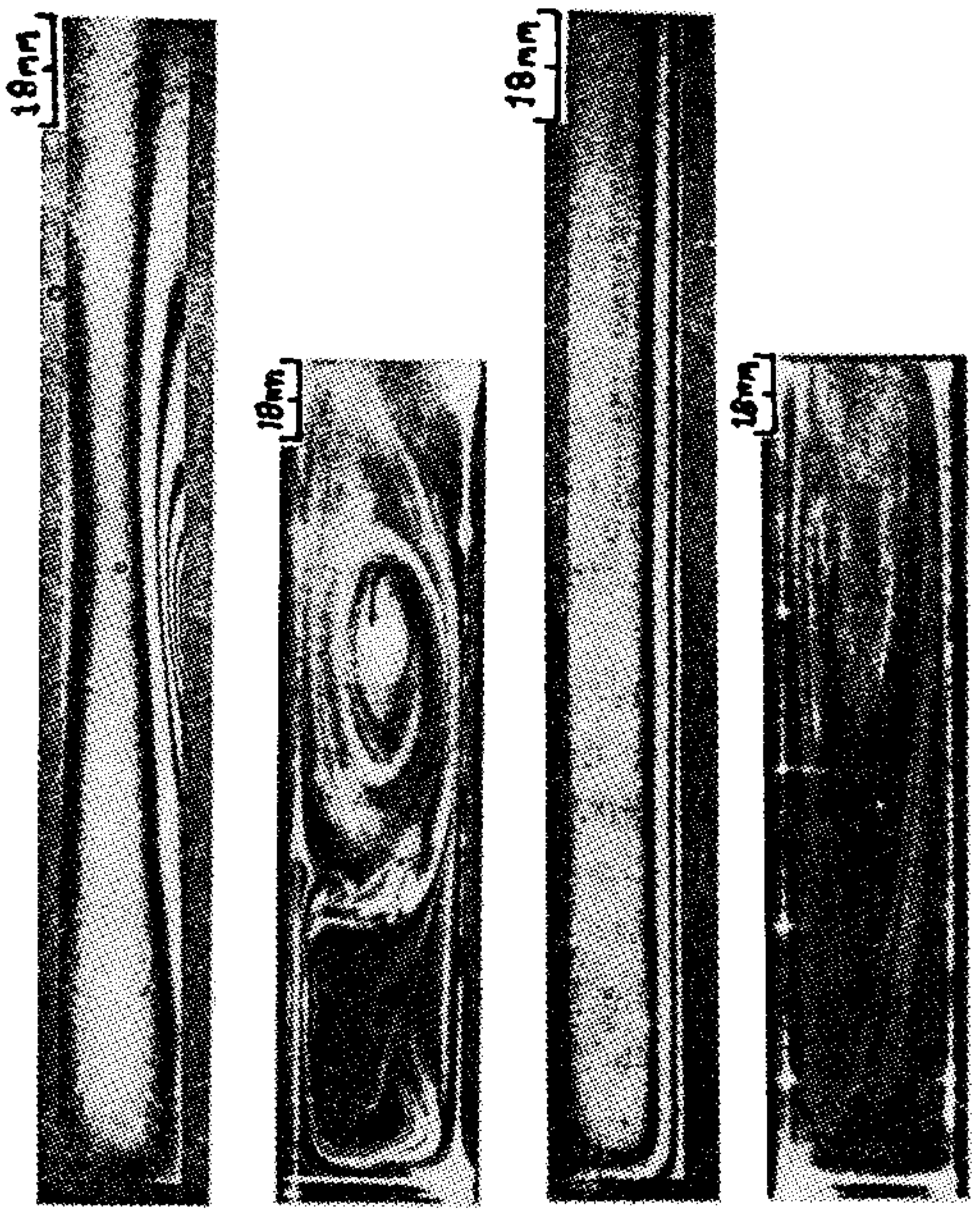
Fig. 2 represents the interferograms of the temperature field in the infinite fringe mode and the typical flow patterns in the cavity for (a) epoxy-resin, (b) copper substrate with flush-mounted heat source for A=9.5, Q_e=2.0W, y/H=0.5.

In Fig. 2(a) for the epoxy-resin substrate, the isotherms show that the hot region are closely spaced at the place where the heat source is located due to low conductivity.

It shows that there is only a little conduction into the vertical wall, so that a lot of the heat must leave in the vicinity of the heat source. A maximum temperature rise 20.6℃ exists at the heated section (y/H=0.5).

At the flow pattern, two recirculating regions are formed. This phenomenon can be expressed as follows. The heated air close to the heat source moves upward to the opening, and then the rising warm air becomes cooled gradually due to heat exchange with the ambient air. Consequently, the cooled air near the opening moves downward following to the continuity. The main flow is stronger than the below one. The below weak recirculating flow is produced because the wall below the heat source is not hot enough to induce the main flow.

In Fig. 2(b) for the copper substrate, the fringes are uniformly distributed along the vertical substrate, and the spacing between the isotherms is almost uniform in the vertical direction due to the high conductivity. In this case, the substrate conduction dominates and the heat transfer occurs mostly through the substrate. It is noticed that the substrate approach an isothermal condition in the limit. The heat conducted into the substrate is



(a) epoxy-resin (b) copper

Fig. 2. An infinite fringe interferograms and typical flow patterns in the cavity for the (a) epoxy-resin, (b) copper substrate with flush-mounted heat source for A=9.5, Q_e=2.0W and y/H=0.5

dissipated from the unheated surfaces to air in the cavity respectively. A maximum temperature rise 10.5℃ exists at the heated section (y/H=0.5). The flow pattern shows that the ambient air entering from the left hand side of the opening moves downward and leaving from the right hand side of the opening. This is because the density of heating air near substrate is smaller than that of the air in the cavity. This flow pattern appears to agree with the predicted one by De Vahl Davis[7] although they used an other thermal conductivity of K=1000 W/mK.

For the same heat source configuration, the maximum temperature rise for the copper substrate is lower than that for the epoxy-resin. This is due to the dominant conduction effect to the substrate on the natural convection.

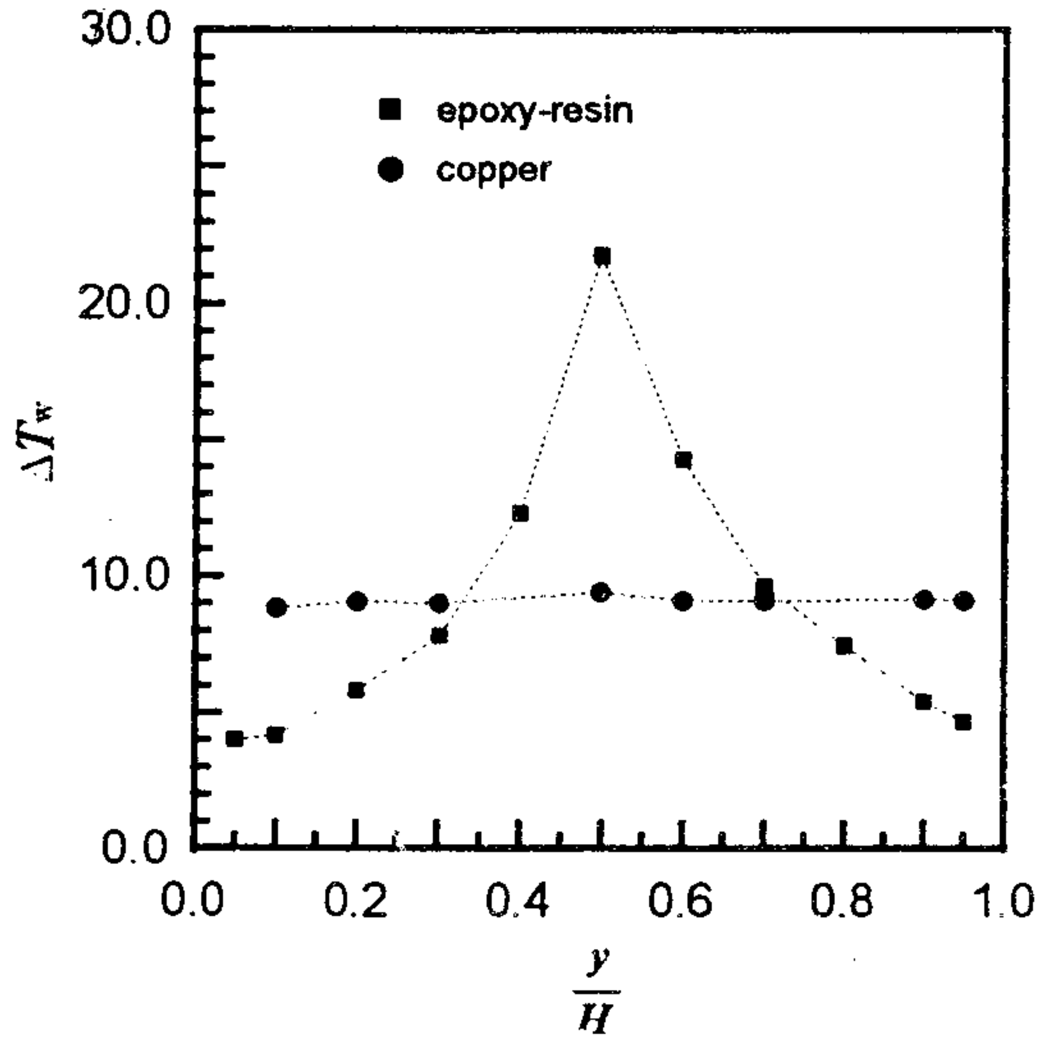


Fig. 3. Local temperature rises along the vertical substrate with flushmounted heat source for A=9.5, Q_e=2.0W and y/H=0.5

Fig. 3 shows the local temperature rises along the vertical substrate with flush-mounted heat source for A=9.5, Q_e=2.0W, y/H=0.5.

For the epoxy-resin substrate, It shows that the local temperature decreases toward the bottom and top of the cavity due to lower conductivity. The maximum temperature rise occurs at the place where the heat source is located. Also, the temperature rises toward the top of the cavity are higher than those for the bottom.

This is because the unheated section above the heat source is affected by the main flow.

For the copper substrate, the temperature is well distributed along the substrate due to the high conductivity.

From Figs. 2 and 3, it is noticed that the cooling effect for the copper substrate is superior to that for the epoxy-resin substrate. Therefore, the use of the highly conductive substrate promises the reduction of the maximum temperature rises.

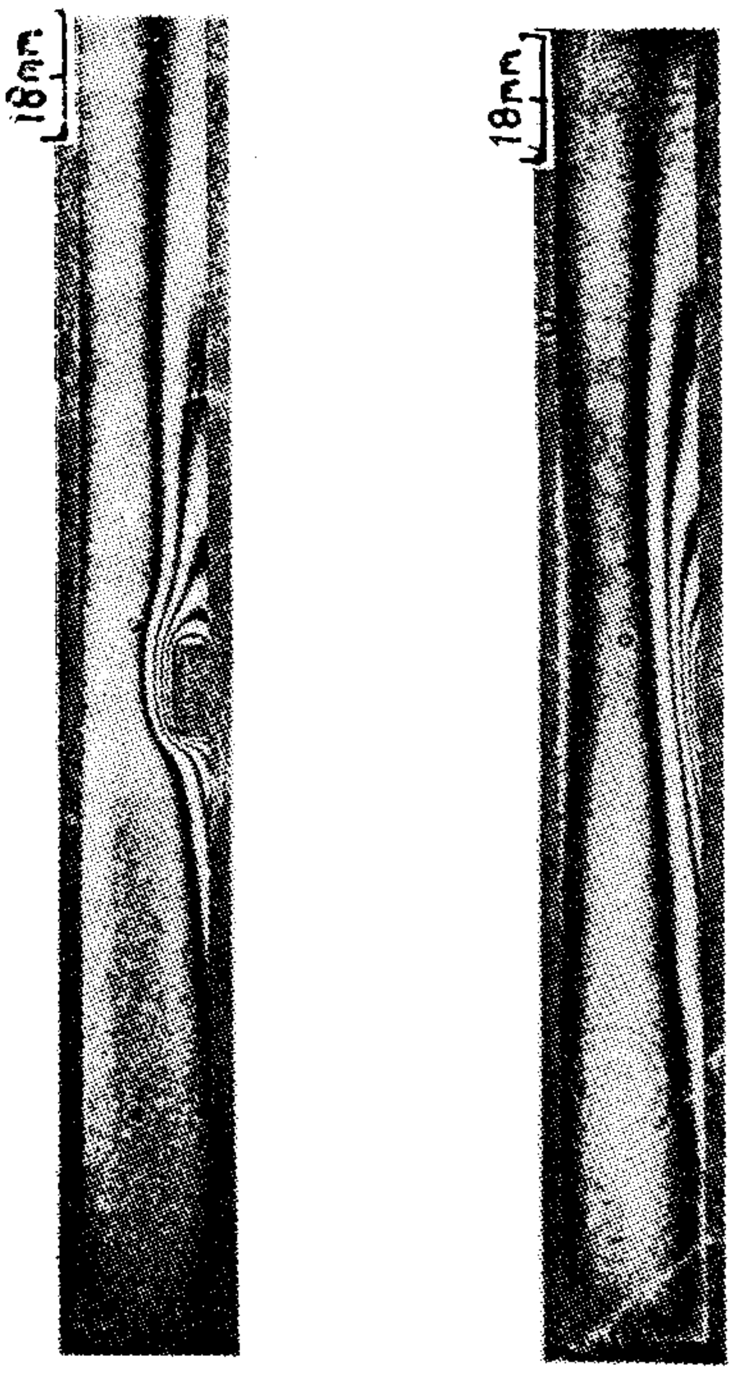
3.2 Effect of the Heat Source Protrusion

Fig. 4 represents the interferograms of the temperature field in the cavity for the epoxy-resin substrate with A=9.5, Q_e=2.0W, y/H=0.5.

Generally, the isotherm patterns show the same trend as the Fig. 2(a). However, at the heated section, the isotherms for the protruding configuration(Fig.4(a)) are closely spaced and the spacing between the isotherms decreases than that for the flush-mounted configuration (Fig. 4(b)).

A maximum temperature rise was 23.8℃ for the flush-mounted configuration, and 30.4℃ for the protruding configuration. So, the maximum temperature rise for the flush-mounted configuration was lower than that of the protruding configuration. Here, the heat transfer from the heat source to the air is significantly influenced by convection coefficient due to the low conductivity. Therefore, in this condition, the protrusion disturbs the convection flow of air.

For the copper substrate, on the other hand, the maximum temperature rise for the protruding configuration is lower than that for the flush-mounted configuration. This is because the overall heat transfer rate is influenced mainly by the effective convection area due to the high conductivity. Therefore, in this condition, the protrusion behaves as a fin.

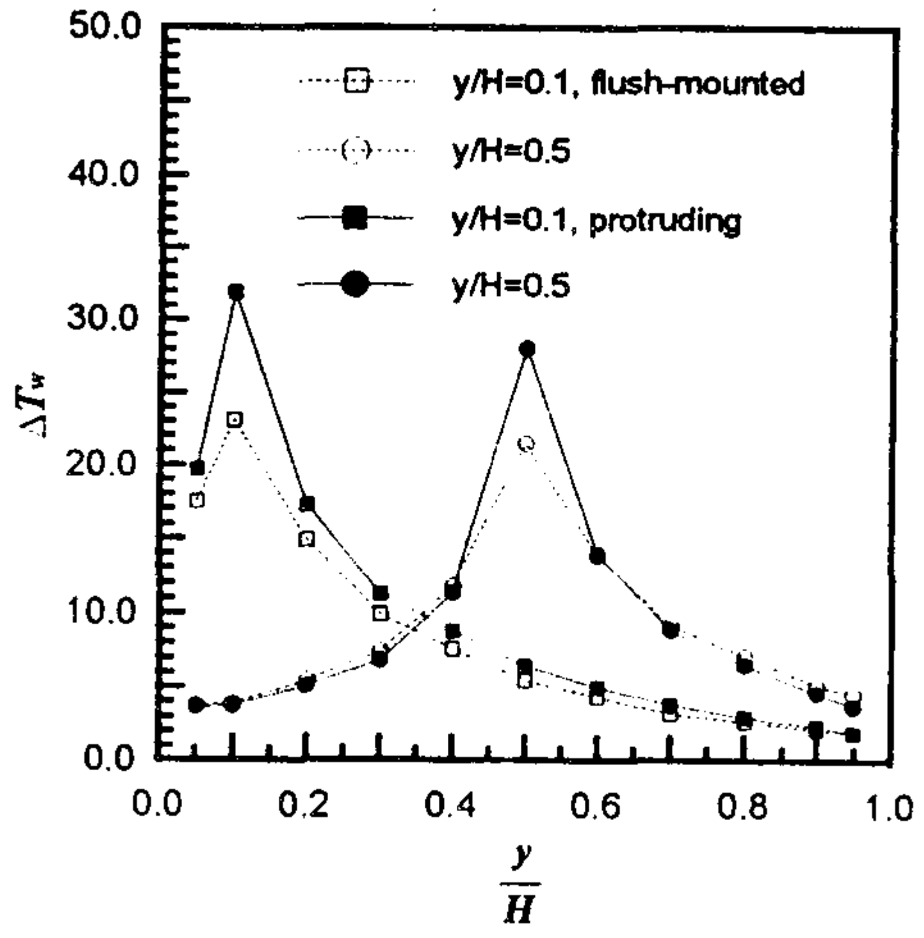


(a) protruding (b) flush-mounted

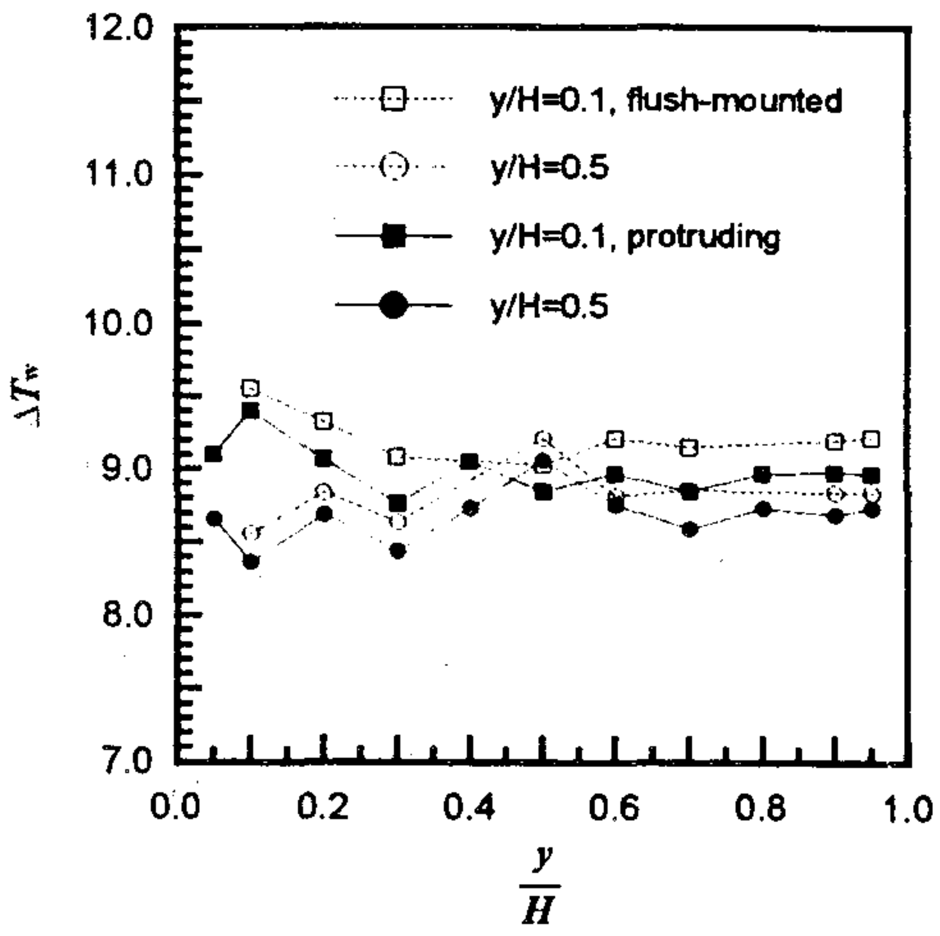
Fig. 4. An infinite fringe interferograms in the cavity for the (a) epoxy-resin, (b) copper substrate with flush- mounted heat source for A=9.5, Q_e=2.0W and y/H=0.5

Fig. 5 shows the local temperature rises along the vertical substrate for the (a) epoxy-resin and (b) copper substrate with A=9.5, Q_c=2.0W, y/H=0.1, 0.5.

In Fig. 5(a), the maximum temperature rise for the epoxy-resin substrate with the flush-mounted heat source is lower than that for the protruding independent of the heat source location. the protrusion of the heat



(a) epoxy-resin substrate



(b) copper substrate

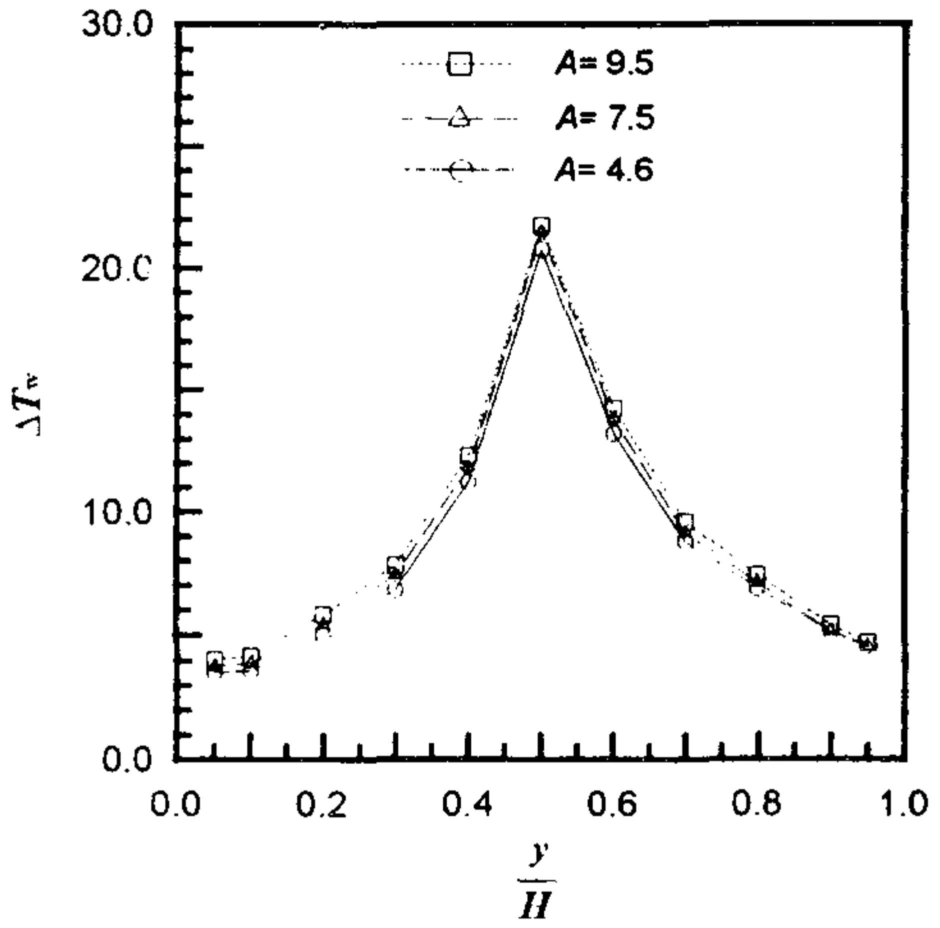
Fig. 5. Local temperature rises along the vertical substrate for the (a) epoxy-resin, (b) copper substrate with A=9.5, Q_e=2.0W and y/H=0.1, 0.5

source plays a role in decreasing the cooling effect.

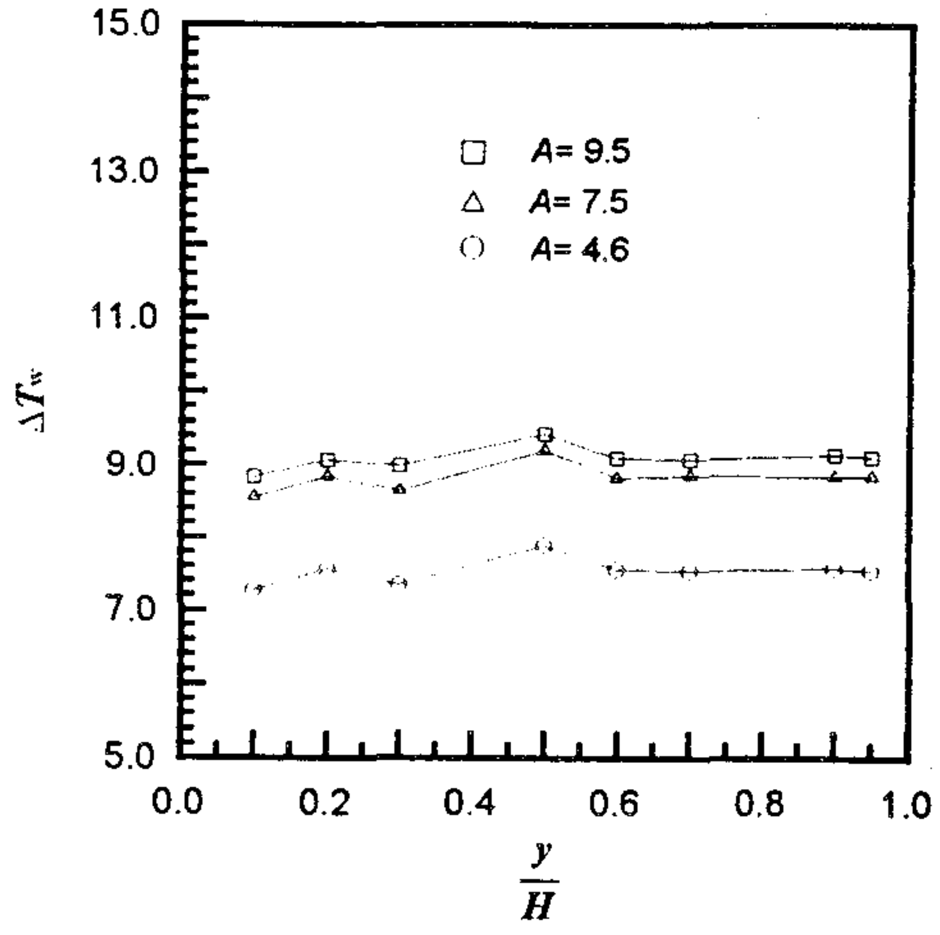
In Fig. 5(b), the maximum temperature rise for the copper substrate with the protruding heat source is lower than that for the flush-mounted independent of the heat source location. In contrast to the case in Fig. 5 (a), the protrusion of the heat source plays a role in increasing the cooling effect.

3.3 Effect of the Cavity Aspect Ratio

Fig. 6 represents the local temperature rises along the vertical substrate of the (a)



(a) epoxy-resin substrate



(b) copper substrate

Fig. 6. Local temperature rises along the vertical substrate for the (a) epoxyresin, (b) copper substrate with flush-mounted heat source for three aspect ratios, Q_e=2.0W and y/H=0.1, 0.5

epoxy-resin, (b) copper with the flush-mounted heat source for three aspect ratios, Q_e=2.0W, y/H=0.5.

The temperature rises tends to increase as aspect ratio increases. The reason is that the convection effect is enhanced due to the increase of mass flow rate entering into the cavity as the cavity becomes wider. The interferograms also showed that the zone of entering fluid in the center portion of the cavity increases in width as the aspect ratio decreases.

3.4 Effect of the Heat Source Location

Fig. 7 represents the maximum temperature rises for different locations of the heat source for A=9.5, Q_e=2.0W.

It reveals that the maximum temperature rises happened at the mid-height of the substrate(y/H=0.5). This result is due to the coupling of conduction and convection heat transfer. At y/H=0.5, the substrate conduction, which is eventually convected to the ambient air in the unheated section, becomes more vigorous due to the effective convection area toward upper and lower surface of the substrate.

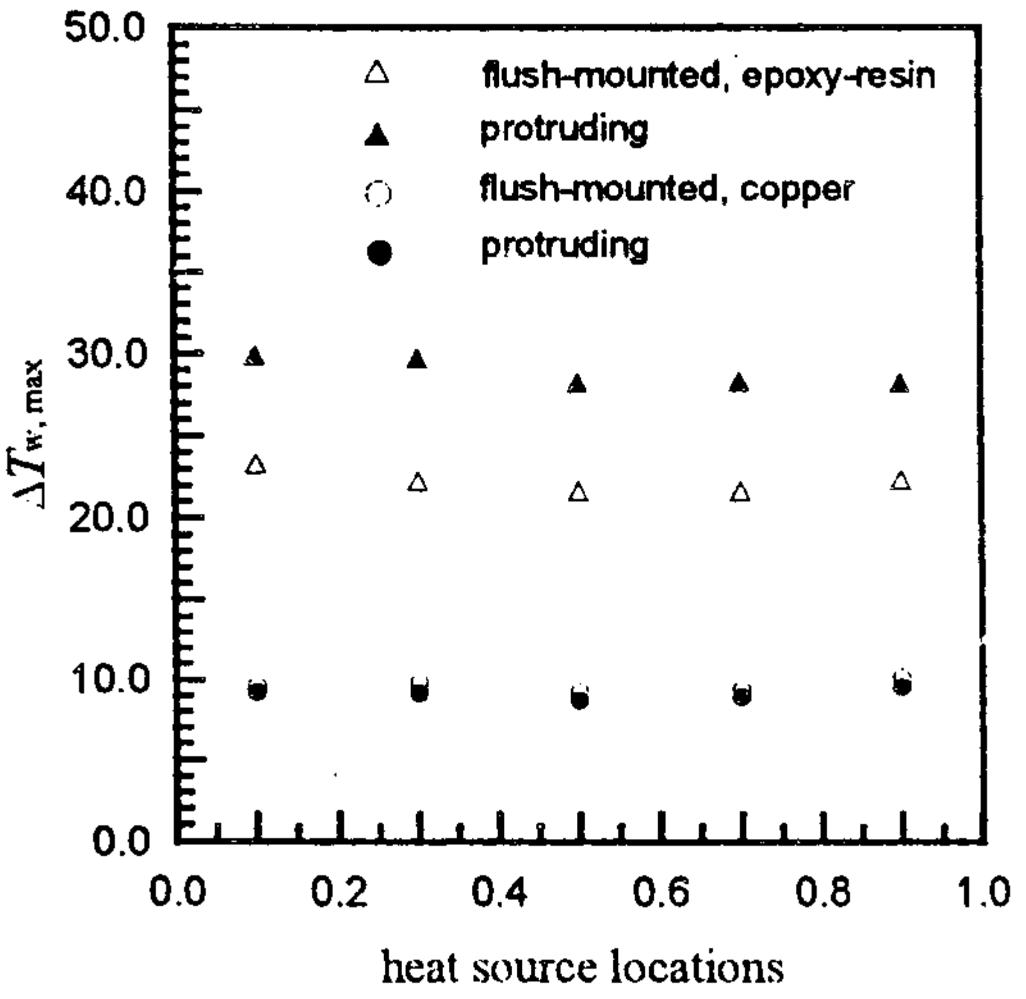


Fig. 7. Maximum temperature rises for different heat source location for A=7.5, Q_e=2.0W

This result is more obvious in the case of the epoxy-resin substrate where almost no conduction occurs to the substrate. The best location is quite independent on the power level considered, which is also observed by Turner[10].

4. Conclusions

The experimental study for two-dimensional steady natural convection in a vertical open top cavity with discrete single heat source have been carried out. The cooling effects for each condition are presented in terms of local and maximum temperature rises on the substrate. Based on the present studies, the following conclusions may be drawn.

The cooling effect for the copper substrate is superior to that of the epoxy-resin substrate and is improved with increasing cavity width. For the epoxy-resin substrate of lower conductivity, the protrusion of the heaters plays a role in decreasing the cooling effect. The beat location was the mid-height of the substrate.

Acknowledgement

The authors wish to acknowledge the

financial support of the Taegu health college.

Reference

1. Underwood, C. I., "Thermal Design and Analysis Spacecraft", UoSAT Spacecraft Engineering Research Unit, Univ. of Surrey, pp.1~40, 1993.
2. Thomas, M., Ruel, C. and Donato, M., "Characterization of a Flat Plate Heat Pipe for Electronic Cooling in a Space Environment", Heat Transfer Electronics Conference, Vol.111, pp. 59~65, 1989.
3. Oosthuizen, P. H. and Kuhn, D., "Unsteady Natural Convection in a Partially Heated Rectangular Cavity", Trans. of the ASME, Vol. 109, pp. 798~ 801, 1987.
4. Jaluria, Y. and Kang, B. H., "Natural Convection Heat Transfer Characteristics of a Protruding Thermal Source Located on Horizontal and Vertical Surfaces", Int. J. of Heat Mass Transfer, Vol. 33, No. 6, pp. 1347~1357, 1990.
5. Churchill, S. W., Patterson, C.V.S. and Chu, H.H.S., "The Effect of Heater Size, Location, Aspect Ratio, and Boundary Conditions on Two-Dimensional Laminar Natural Convection in Rectangular Channels",Trans. of the ASME, pp.194~201, 1976.
6. Richards, D. E. and Ravine, T. L., "Natural Convection Heat Transfer from a Discrete Thermal Source on a Channel Wall", Trans. of the ASME, Vol. 110, pp. 1004~1006, 1988.
7. De Vahl Davis, G. and Behnia, M., "Natural Convection Cooling of an Electronic Component in a Slot", Proc. of 9th Int. Heat Transfer Conference, Vol. 3-EL-11, pp. 343~348, 1990.
8. Riu., K. J., Choo, H. L. and Oh, S. Y., "Transient Natural Convection Cooling in a Vertical Open Top Cavity with Discrete Heat Sources", Proc. of KSME Annual Meeting, Vol. 2, pp. 61~66, 1995.
9. Hauf, W. and Grigull, W., Advances in Heat Transfer, Vol. 6, pp. 133~339,